NISTIR 6242

ANNUAL CONFERENCE ON FIRE RESEARCH
Book of Abstracts
November 2-5, 1998

Kellie Ann Beall, Editor

Building and Fire Research Laboratory
Gaithersburg, Maryland 20899

United States Department of Commerce
Technology Administration
National Institute of Standards and Technology

NISTIR 6242

ANNUAL CONFERENCE ON FIRE RESEARCH
Book of Abstracts
November 2-5, 1998

Kellie Ann Beall, Editor

October, 1998
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

U.S. Department of Commerce
William M. Daley, *Secretary*
Technology Administration
Gary Bachula, *Acting Under Secretary for Technology*
National Institute of Standards and Technology
Raymond G. Kammer, *Director*

MITIGATION OF COMPARTMENT JET FIRE USING WATER SPRAY

K Alageel, BCR Ewan, J Swithenbank
Department of Chemical and Process Engineering, University of Sheffield, Mappin Street, Sheffield, S1 3JD, UK

INTRODUCTION

The main objective of the study is to investigate the interaction of the water sprays with a jet fire in an underventilated (confined) space, using CFD. In order to achieve the above objective, it is necessary to produce a workable CFD model of a confined jet which is representative of conditions which might exist offshore.

For this purpose a compartment has been chosen since this represents a geometry which has been the subject of a number of experimental and analytical studies.

COMPARTMENT GEOMETRY AND OPERATING CONDITIONS

The compartment dimensions were those of a standard container being 6m in length by 2.4m high and 2.4m in width. In order to extend the computation space beyond the actual compartment, in the manner recommended for example by Markatos (1), the compartment was situated within a larger chamber, allowing the cells at the compartment opening to be live. This outer environment provided the source for combustion air and an outlet for all the gases to exit, without imposing restrictions to the flow into and out of the compartment.

Propane gas is injected into the compartment through a 1.5 cm diameter nozzle of length 25 cm situated on the floor of the compartment and at its centre. Air is provided from the outer environment chamber into the region of the compartment opening and at low velocity to enable the compartment to entrain as much as is required for combustion.

The propane gas jet entrains air into the compartment, and so the flow rate of air into the domain defined by the outer chamber needed to be several times the stoichiometric quantity needed for combustion. This outer air supply flow velocity was around 0.3m/s and the propane velocity at the jet nozzle was equal to 250 m/s.

TWO-PHASE MODELLING

In order to combat fire it is necessary to understand the nature of the interaction between the hot combustion products and the liquid water. Factors which need to be considered include water flow rate, spray pattern, droplet size and the number and location of spray heads.

It is important also to find out the maximum amount of water that can be discharged from an appropriate spray head in order not to flood the compartment and cause water damage.

The simulation started by studying the effects of a single spray located at different positions in the roof of the compartment. This will give a chance to better understand the fire-spray interaction and evaluate the best spray(s) location to be used to carry out a more detailed investigation. The total water flow rate for each of these arrangements was varied from 0.1 to 3.3 kg/s and the velocities of the droplets used varied between 5 and 25 m/s. The mean droplet diameters could be chosen within the range from 100 to 600 μm. Finally the spray angle used for each of the spray heads is represented by 5 injection directions with each direction having the possibility of an independently defined size and velocity range.

Three spray locations were examined. In the first, a single spray is located above the propane nozzle. The remaining two placed on the front/rear axis and 2.0m on each side of the centre.

RESULT AND DISCUSSION

Initially, the steady state behaviour of the compartment fire was evaluated and used as the starting condition for the subsequent two phase calculation. Prior to the spray activation, the fire plume was able to rise straight upwards and spread outwards along the ceiling. For the case of a single spray located above the propane jet and at a low flow rate less than 1 kg/s, two major flows were apparent after the solution fully developed. The first, generated by the water spray, was downwards whilst the second, generated by the fire, was along the ceiling. These two currents met towards the centre of the compartment, aiding the mixing and cooling process.

The value for temperature used in the calculations throughout this paper, is the average temperature from the upper two-third of the compartment. Because of the configuration of the inlet to the compartment, realistic values of temperature are only obtained above this cooler inlet region.

From the modelling of different spray locations, the results show that less water is needed to extinguish the flame in the case of a single spray located centrally above the jet nozzle. For most flow rate, this location also gave lowest temperatures as shown in Figure 4. Subsequent modelling therefore used a single spray located in the centre of the compartment above the jet nozzle.

Figure 1 shows the average temperature in the compartment as function of mean droplet diameter of 100, 200, 300, 400, 500 and 600 μm. The curve is nearly parabolic and the lowest temperature was found with mean droplet diameters of 300 μm. Subsequent modelling was therefore carried out using mean diameters of 300 μm. It is likely that the minimum arises from the competing effects of droplet penetration which increases with droplet diameter and total droplet surface area for evaporation.

Modelling different spray angles of 30, 60, 75, 90, 100, 120, 135 and 150° indicated that using spray angle of 60° and 75°

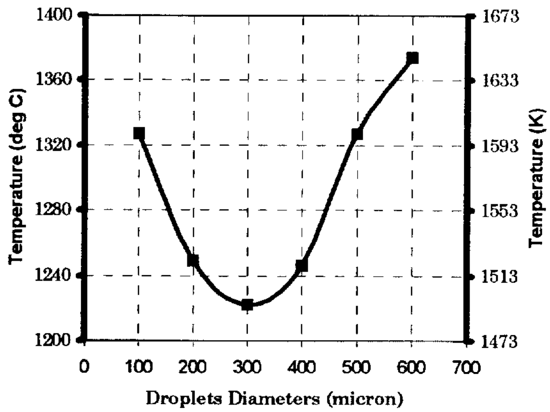

Figure 1. Comparison of different droplets diameters.

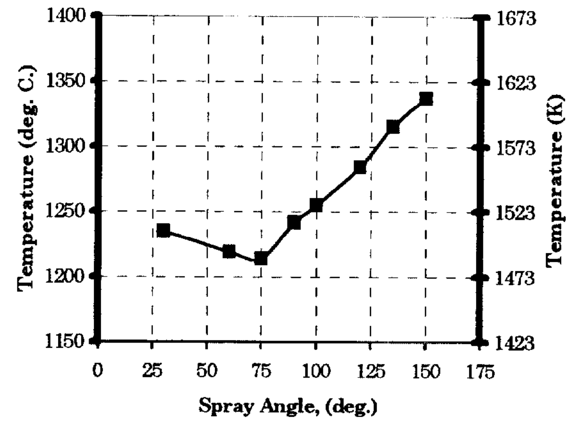

Figure 2. Comparison of different spray angles.

was most effective in reducing the overall temperature as can be seen in Figure 2. Subsequent examination of the effect of spray droplet velocity has therefore used this optimised 60° spray angle. The effect on average temperature of varying this velocity from 5-25m/s is shown in Figure 3. This shows the limiting behaviour due to the effectiveness in penetrating the flame and indicates that for this geometry, velocities in excess of 18 m/s should be used.

For those cases where the water flow rate produced large reduction in average temperature, it was considered that the assumption of steady state was invalid since extinguishment was the likely outcome. For these cases, a time dependent calculation was performed which has the capability of predicting the change in combustion variables with time during this process.

The time dependent simulation was performed with a 1 second time step and during each of these steps, iterations across the domain were carried out until convergence criteria of the variables solved were satisfied. Further details of these results can be found in reference (2).

CONCLUSION

The water discharge rate, the median drop size of the water spray, the spray angle of injections, the droplets velocity, and the heat release rate of the fire source were recognised as important parameters in the cooling of compartment fires by water spray. Time dependent calculations provide a valuable insight into the changes taking place during the extinguishment process. It is clear that present CFD codes with combustion are now able to provide us with details of the factors which control the steady state behaviour of confined fires and emphasise the importance of a mechanistic approach to the study of the dynamics of extinguishment.

REFERENCE

1. Markatos N.C., Malin M.R. and Cox G., Int. Journal of Heat Mass Transfer; Vol. 25 (1); pp.63-75. 1982
2. Alageel, K., Ewan, BCR and Swithenbank, J., IChemE Research Event, Newcastle Upon Tyne, UK, 7-8 April 1998.

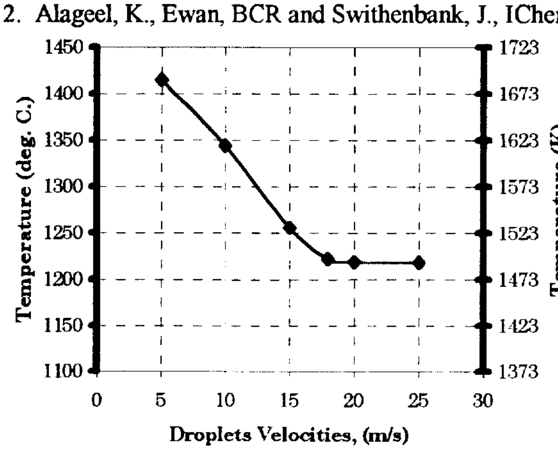

Figure 3. Comparison of different droplets velocities.

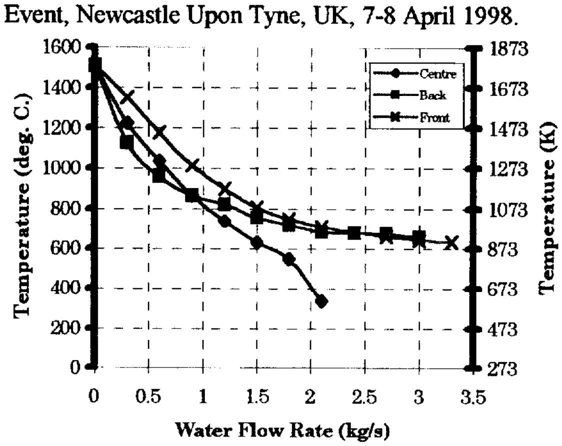

Figure 4. Comparison of different flow rate for different locations.